NASA/TM—2000-209952

Challenges in Integrating Nondestructive Evaluation and Finite Element Methods for Realistic Structural Analysis

Ali Abdul-Aziz
Cleveland State University, Cleveland, Ohio

George Y. Baaklini
Glenn Research Center, Cleveland, Ohio

Dmitri Zagidulin
Allegheny College, Meadville, Pennsylvania

Richard W. Rauser
Cleveland State University, Cleveland, Ohio

April 2000

Since its founding, NASA has been dedicated to the advancement of aeronautics and space science. The NASA Scientific and Technical Information (STI) Program Office plays a key part in helping NASA maintain this important role.

The NASA STI Program Office is operated by Langley Research Center, the Lead Center for NASA's scientific and technical information. The NASA STI Program Office provides access to the NASA STI Database, the largest collection of aeronautical and space science STI in the world. The Program Office is also NASA's institutional mechanism for disseminating the results of its research and development activities. These results are published by NASA in the NASA STI Report Series, which includes the following report types:

- TECHNICAL PUBLICATION. Reports of completed research or a major significant phase of research that present the results of NASA programs and include extensive data or theoretical analysis. Includes compilations of significant scientific and technical data and information deemed to be of continuing reference value. NASA's counterpart of peer-reviewed formal professional papers but has less stringent limitations on manuscript length and extent of graphic presentations.

- TECHNICAL MEMORANDUM. Scientific and technical findings that are preliminary or of specialized interest, e.g., quick release reports, working papers, and bibliographies that contain minimal annotation. Does not contain extensive analysis.

- CONTRACTOR REPORT. Scientific and technical findings by NASA-sponsored contractors and grantees.

- CONFERENCE PUBLICATION. Collected papers from scientific and technical conferences, symposia, seminars, or other meetings sponsored or cosponsored by NASA.

- SPECIAL PUBLICATION. Scientific, technical, or historical information from NASA programs, projects, and missions, often concerned with subjects having substantial public interest.

- TECHNICAL TRANSLATION. English-language translations of foreign scientific and technical material pertinent to NASA's mission.

Specialized services that complement the STI Program Office's diverse offerings include creating custom thesauri, building customized data bases, organizing and publishing research results . . . even providing videos.

For more information about the NASA STI Program Office, see the following:

- Access the NASA STI Program Home Page at *http://www.sti.nasa.gov*

- E-mail your question via the Internet to help@sti.nasa.gov

- Fax your question to the NASA Access Help Desk at (301) 621-0134

- Telephone the NASA Access Help Desk at (301) 621-0390

- Write to:
  NASA Access Help Desk
  NASA Center for AeroSpace Information
  7121 Standard Drive
  Hanover, MD 21076

Challenges in Integrating Nondestructive Evaluation and Finite Element Methods for Realistic Structural Analysis

Ali Abdul-Aziz
Cleveland State University, Cleveland, Ohio

George Y. Baaklini
Glenn Research Center, Cleveland, Ohio

Dmitri Zagidulin
Allegheny College, Meadville, Pennsylvania

Richard W. Rauser
Cleveland State University, Cleveland, Ohio

National Aeronautics and
Space Administration

Glenn Research Center

April 2000

Available from

Challenges in Integrating Nondestructive Evaluation and Finite Element Methods for Realistic Structural Analysis

Ali Abdul-Aziz*
Cleveland State University
Department of Civil and Environmental Engineering
Cleveland, Ohio 44115

George Y. Baaklini[†]
National Aeronautics and Space Administration
Glenn Research Center
Cleveland, Ohio 44135

Dmitri Zagidulin[‡]
Allegheny College
Meadville, Pennsylvania 16335

Richard W. Rauser[§]
Cleveland State University
Department of Civil and Environmental Engineering
Cleveland, Ohio 44115

Abstract

Capabilities and expertise related to the development of links between nondestructive evaluation (NDE) and finite element analysis (FEA) at Glenn Research Center (GRC) are demonstrated. Current tools to analyze data produced by computed tomography (CT) scans are exercised to help assess the damage state in high temperature structural composite materials. A utility translator was written to convert velocity^2 (an image processing software) STL data file to a suitable CAD-FEA type file. Finite element analyses are carried out with MARC, a commercial nonlinear finite element code, and the analytical results are discussed. Modeling was established by building a MSC/Patran (a pre and post processing finite element package) generated model and comparing it to a model generated by Velocity^2 in conjunction with MSC/Patran Graphics. Modeling issues and results are discussed in this paper. The entire process that outlines the tie between the data extracted via NDE and the finite element modeling and analysis is fully described.

Introduction

Most reverse engineering approaches involve an object imaging or digitizing and then creating a computerized reconstruction that can be integrated, in 3D, into a particular design environment. Rapid prototyping (RP) refers to the practical ability to build high-quality physical prototypes directly from computer aided design (CAD) files. Using rapid prototyping, full-scale models or patterns can be built using variety of materials in a fraction of the time required by more traditional prototyping techniques [1,2].

Many software packages have been developed and are being designed to tackle the reverse engineering and rapid prototyping issues mentioned above. For instance, Velocity^2 [3] is being used to carry out the construction process of 3D volume models and subsequent generation of a stereo-lithography (STL) file that is suitable for CAD application. Producing three-dimensional models of objects from the CT scans is becoming a valuable Nondestructive Evaluation methodology [4]. Real components can be rendered and subjected to temperature and

stress tests using structural engineering software codes. To achieve this, accurate high-resolution images have to be obtained via CT scans, and then processed, converted into traditional file format, and translated into finite element models.

FEA has been used extensively to model the effects of static and dynamic loading on aerospace propulsion components. This technique allows the application of complicated loading schemes by breaking the complex part geometry into many smaller, geometrically simple elements. Commercial codes such as MARC [5] are used extensively to determine the structural characteristics of a given component under various loading conditions.

Therefore, the main objectives of this work is to clearly apply the most advanced techniques available in image processing to develop a comprehensive methodology connecting NDE with FEA. In addition, this will include performing analytical studies to evaluate the structural integrity of aerospace components under complex loading conditions.

Image Processing and 3D Rendering

This section of the report is devoted to describing and detailing the steps followed in constructing an object 3D image and the subsequent finite element model. Several sub-paragraphs are listed below and each outlines the procedure applied, the pros, the cons of the software used, and their capabilities.

1. Image Quality and Alignment

The image quality is a crucial aspect of the 3D rendering and modeling process. The accuracy of the model depends on the clarity and the level of detail of the original CT scans from which it will be produced. While filters and image-editing tools that exist help sharpen and clean up the images, none of these can make up for a lack of quality and high-resolution originals.

During the process of scanning the object to be modeled using CT, the distance between the slices is brought to a minimum, (i.e. a slice every millimeter or smaller), since this increases the accuracy of the subsequent model. This approach usually generates more files and hence more information to process. However, it is necessary in order to improve the accuracy and to complete the construction of the model. In the current work, only four image slices of the panel were available. The rendering software, which fills in the missing information, is used to extrapolate between these slices and to create more images for the 3D-model construction.

Figure 1 is a representative CT scans slice of a cooling panel that is the component of interest in this work. It consists of composite material and four metal tubes brazed to the composite by a brazing alloy to ensure adherability to the composite. Ideally, the CT scan images would be "optically aligned" – that is, aligned to each other by the scanning process itself, with all the slices having the same dimensions and orientation. However, this was not the case with the available slices, so the images had to be manually aligned to each other, using specialized software, VayTek's *Slice Align* [6]. Figure 2 shows the aligned, pre-processed and filtered image of the CT scan shown in Figure 1.

2. Pre-Processing

Most of the time, Computed Tomography images will have speckles, interference, and artifacts caused by the scanning process itself. Using image-editing software, it is possible to reduce or eliminate these unwanted artifacts. Some 3D reconstruction software packages include their own filtering and image editing options, but conventional photo-editing packages such as Corel PhotoPaint [7] or Adobe PhotoShop [8] turned out to have the most convenient features for image pre-processing. Both the software packages mentioned were used to improve the image quality of the slices and to guide in completing the 3D construction of the model.

3. Image Editing

To remove the artifacts in the image, each individual image/slice was handled separately. Since these artifacts will differ, however slightly, with each image, there are no automated ways to work on all the images at once. Removing the artifacts involves "turning off" or erasing the pixels that comprise them. Velocity2 provided a limited capability for artifact removal through its use of masks. A "mask" is created from the whole image, and the individual pixels of the mask can be erased. Note that this does not erase the pixels on the original image, only on the mask. The mask, not the image, is then used for 3D rendering. The difficulty with this process is that the mask is of one color (e.g. white or yellow), and so it's difficult to determine where the legitimate object ends and the artifact begins when erasing.

So editing the images was implemented by a more convenient alternative to artifact removal, which is provided by the image editing software, Corel PhotoPaint [7]. Using image-editing tools such as Erase and Color Change, the image was edited and updated from all the artifacts.

Furthermore, while editing the image, it is sometimes necessary to add and to remove objects by drawing-in components that were not picked up by the CT scan, or to mark the image for future use. In this paper case here, the cooling panel being modeled consisted of fiber reinforced composite material and four metal tubes. Only the metal of the tubes and the brazing material that was used to hold the metal tubing to the composite showed up on the CT scan – the composite material was much less dense. Therefore, to simulate the existence of the composite material, a manual operation using one of the available CAD software packages was applied. Velocity^2 has no CAD skills and image editing programs like Photopaint [8] and drawing programs like Corel Draw [8] offered none either. MSC/Patran [9] was the only tool available to provide such capabilities.

4. Filtering and Grayscale Operations

Improving the qualities of the images by removing noise, sharpening the edges, and enhancing the CT scans is recommended. Using options like filters and grayscale are known to enhance the image and to improve its graphical features. For instance, filters are complex operations applied to the entire image, altering each pixel in a way that depends on its surrounding pixels. While Grayscale operations are simpler, and they affect each pixel on the image regardless of its neighbors. An appropriate filter or operation can be performed on the entire set of images, one by one. Also, scripts can be recorded to provide an automated way to apply several filters to all the images in the series. Filters such as Median, scale, Logarithm and sharpen were all applied to upgrade the quality of the images and make them easier for rendering. However, caution must be exercised since excessive use of filters and grayscale operations will lead to loss of information. Figure 3 shows the 3D-rendering model of a set of 2D filtered CT slices.

5. Regions of Interest (ROIs)

Most of all the 3D rendering and image processing software possess means to define and isolate regions of interests (ROI). For example, when modeling the cooling tubes, it is desirable to render the brazing material around the tubes separately, to later provide that region with a different set of material characteristics. Therefore, to capture the later effects, the brazing material which appears as a bright region on the images is isolated by using Velocity^2 's Logic Operations: **Flood Fill All**. This is done by setting the brightness threshold to an appropriate level that will only highlight the brazing and not the rest of the tubes. If no appropriate setting can be found, another way to isolate this region of interest would be to highlight the brazing, pixel by pixel. Once highlighted, the region can be rendered separately, and then imported into MSC/Patran along with the tubes and given separate material properties. Similarly, Velocity^2's Logic Operations: **Flood Fill All** is used to identify the metal tubes and the composite.

6. 3D Reconstruction

Velocity^2 is used to complete the process and to construct the actual 3D model. The procedure followed extrapolating the 2D series of images generated via the aforementioned mini-procedures into a 3D view. This is included creating an **.info** file and verifying the sharpness of the images before beginning the rendering process. Since the edge-detection and rendering algorithms will need this data for accurate modeling. It is worth mentioning

that the most difficult part of 3D modeling is the image pre-processing and the ROI isolation. Upon completing these steps, the actual 3D reconstruction is no more than a menu driven process. In Velocity^2, invoking the **Surfer Tab** option, selecting the output file, the material, and setting the threshold would provide the necessary sequence to run the 3D construction.

7. Polygon Rendering

Upon completion of the 3D rendering, a resulting file that contains the geometric information needed for constructing a three dimensional finite element model is generated. The resulting binary file must undergo several more transformations before it's ready for importation into a CAD program such as MSC/Patran [9] or Hypermesh [10]. In Velocity^2, the rendered file is stored as Stereolithographic (STL) format file. This format exists for easy transfer between Velocity^2 and other applications, including HyperMesh and the STL-To-Patran, (PNF), an in-house translator. More details regarding the functionality of this translator are offered in sub-paragraph 10 (STL translator).

It is important to keep in mind that the polygon rendering provides only the surface of the objects, that is, a wireframe mesh of the objects with no elements inside. This is sufficient for some applications. However, to perform any analytical task and to assign material properties to the objects, the model has to be modified and to be setup for analysis by using specialized solvers such as HypeMesh [10] or MSC/Patran [9]. Figure 4 illustrates the polygon-rendering format of a section of the panel.

8. Polygon Reduction

One of the major problems in 3D modeling is the sheer number of polygons that the rendering and, more importantly, the analysis software have to deal with. For relatively simple objects to be adequately modeled, hundreds of thousands of triangular elements, and, consequently, a large number of nodes are needed. Reducing these numbers without losing model accuracy and detail is a considerable challenge that is yet to be completely overcome. Ultimately, one must arrive at a compromise – reduce the number of polygons as much as one's tolerance for accuracy allows, and have enough computing power to work with the remaining ones.

Two important tools to reduce polygon count are Velocity^2 's PolyMerge [6] which reduces the polygon counts of rendered files (before converting them into STL files), and Decimator [11], a utility that reduces and simplifies the polygons in STL files. It was determined that PolyMerge can be run without significant loss of detail with settings of 10-25 percent. Both utilities are needed to produce manageable models. When reducing, one should keep an approximate polygon count goal in mind, and then adjust the PolyMerge [6] and Decimator's [11] settings to achieve it. Thus, if one's machine can handle 100,000 elements comfortably, one should run PolyMerge [6] at a reasonable setting, for example a 20%, and then use Decimator [11] to reduce the remaining polygon count down to 100,000. Figures 5 and 6 show both the polygon wire mesh and the IGES model of one tube section of the panel, respectively. The polygon wire mesh was reduced to 35,000 polygons down from 100,000 polygons before reduction.

9. Volumetric Rendering

Polygon rendering of objects, using Velocity2 or VoxBlast only produces "hollow" surfaces. In addition, it is desirable to work with "solid" objects possessing material properties when performing finite element analysis. As a result, one must volume-render STL files using either HyperMesh [10] or MSC/Patran [9] depending on the application thought. Automation of this procedure is an ongoing task and this work represents the groundwork for it.

10. STL Translator

Velocity^2 's output which is the polygonized 3D volume rendered is written in a common format called stereolithographic) (STL). However, this format is not suitable to be read by MSC/Patran. Therefore, a software translator to convert the STL file into a neutral file interfaceble with MSC/Patran was developed. The code was written in C language and it can recognize both ASCII and binary format. The code is called PNF translator where

PNF stands for Patran Neutral File. The code was complied to be used on Silicon Graphics platform, however, it can be compiled on any other machines by following the instructions for a particular C++ compiler.

Analytical Procedure

The analyses conducted in this study constitute a major goal and a completion of the image-rendering scheme employed. As stated before, it is intended to generate a connection between the main image rendering software Velocity^2 and the CAD applications software MSC/Patran or Hypermesh. The STL files produced by Velocity^2 are recognizable by Hypermesh and an MSC/Patran neutral file can be easily disposed from Hypermesh as well. However, Velocity^2-Patran link is only possible through the in-house PNF translator. For instance, in this work Hypermesh and MSC/Patran were used concurrently to construct the object finite element model.

Furthermore, upon translating the STL file into polygons and wire mesh, a rigorous editing of the wire mesh remains. This is to eliminate and to reduce the number of surfaces in the model. Only tetrahedral and triangular elements can be generated with HyperMesh, while hex elements and others are possible with MSC/Patran.

Finite Element Analysis

The first part of the analysis covered generating a finite element model for the cooling panel using the geometric entities provided by the CT scan image with MSC/Patran. The panel's dimensions were 0.3225 in (0.82 cm) thickness, 1.349 in (3.426 cm) length and 1.496 in (3.8 cm) for the width. The three dimensional model consists of 25025 hex element eight node and 20920 nodes, Figure 7. It is built purposely large to capture as many details as possible and to insure that the modeling resembled the as manufactured configuration of the panel. It was made out of fiber reinforced composite material, and metal tubing, 3/16 in (0.47625 cm) outside diameter, with a wall thickness of 0.035 in (0.0889 cm) and brazing molding alloy of 0.05 inch (0.127 cm). Figure 8 illustrates the materials distribution in the model.

Figure 4 represents the polygon rendering format model of the CT scan image developed via the procedure described in this paper. This is the production of the CT scan images processed with Velocity^2. However, this model was not developed further to be used in the analyses due to its complex structural shape and large size. It was beyond the capability of both the finite element and the CAD codes available. As a result, only one section of the panel, which included one tube, was modeled for the analysis. Figure 9 shows the finite element model of the one tube section of the panel including the composite. It consists of 8860 eight node hex elements and 10737 nodes. Note that the geometric irregularities captured by the CT scan on the model are obvious. For instance, the tube circular section is not as smooth. Deformities caused by the combined coolant high pressure and the thermal loads are clearly marked. The brazing material around the metal tubing acquired an irregular shape. Similarly, an MSC/Patran version of the one tube section was generated and it is represented in Figure 10. This is to enable comparing CT scan data finite element. It consists of 840 eight-node elements and 1176 nodes.

The finite element analyses were performed on the cooling panel under elastic conditions and combined thermo-mechanical loading. The panel was subjected to high-pressure coolant injected through the cooling holes and to a thermal loading as well. The thermal load was assumed isothermal. The panel was constrained in a fashion that the bottom side would have a zero displacement along the vertical direction to simulate gripping effects invoked during testing, Figure 7. While in Figure 9 which represents the single tube case additional constrains was added by fixing the edge nodes in the X-direction. Rigid body motion was assumed by fixing one single node in all directions in both cases. The panel was free move along the Y-axis. Material properties for the panel were acquired from reference [12]. The composite and brazing alloy possess orthotropic material characteristics. Such material effects were all incorporated into the analyses.

The panel underwent broad test matrix investigating its thermal performance, thermal shock performance and thermo-mechanical life cycle. However, the current work is only investigating identifying the structural deformities and the effects of the brazing material based on NDE data. Since it is known that the brazing material might erodes

the metal tubing during the processing cycle, especially at the access holes where the brazing could overflow. The applied loading covered thermal and mechanical loading due to the coolant high pressure.

The stress analyses were conducted with MARC code [5] and results produced are discussed fully in the subsequent section.

Discussion of Results

The methodology applied here to connect the NDE data with FEA is still in the development stages, a more robust procedure is in the works to improve the sequence of generating the data and setting up the image model. Analytical evaluation and structural assessment are of interest and they represent the ultimate goal. For instance, the size of the model developed from the CT images remains very large. And the software utilized to construct it is still not equipped with the tools to reduce it as desired without jeopardizing the accuracy of the model. The challenge remains in resizing the geometry to an acceptable level.

In spite of the above, the overall purpose in this paper is to demonstrate the applicability of the finite element in tracing NDE modalities. Such as modeling images generated by MRI's, CT Scans, X-ray's etc into meaningful structural entities. The procedure applied in demonstrating these capabilities is outlined in details in the beginning of the paper. A clear representation of the sequence or the route carried out to construct a three dimensional volume from a set of 2D images and convert it into a finite element model ready for analyses is documented.

CT density data and stiffness data were not incorporated into the finite element mesh developed due to the unavailability of a computational scheme that correlates the two. Such scheme is undergoing development and it has been tested for tetrahedral elements [13]. However, it has not materialized to be a complete and robust procedure. It still lacks the capability to handle large models and to identify deformities and obvious cracks detected by CT scans. Therefore, for simplicity and for the purpose of this report, material density reductions were kept constant within a material system. Based on the imaging arrangement in Velocity^2 and the CT data, the material systems were identified as the metal and the brazing alloy and the fiber reinforced composite.

Results obtained from the finite element analyses are shown in Figures 11 to 13. Figure 11 shows the stress distribution in the entire cooling panel. It is clearly noted that the tubes experience the highest stresses, which are due to the combined effects of both the thermal and the coolant high-pressure loads. Such high stresses may have caused the brazing material surrounding the metal piping wall to deform. This scenario could easily promote leaky tube conditions. In addition, it might cause the brazing to be forced through the access holes, not included in the modeling. CT scan findings have confirmed such possibilities. Nevertheless, the composite material enjoyed a lower stress magnitude compared to both the metal tubing and the brazing materials.

Figure 12 shows the stress distribution reported in the one tube section of the cooling panel using the CT scan developed model. Once more, the maximum stress location is at the metal tube inside wall, which is similar to that experienced by the data shown in Figure 11. However, in this case the brazing material deformation is clearly noted due to the cooling gas high pressure. Such data were noticed in the CT scan and interpretation of the information collected confirmed failure and leaking at the metal-wall-brazing material interface. Moreover, the finite element succeeded in identifying these behaviors. Furthermore, Figure 13 is a representative of the stress distribution generated by the finite element analysis on the MSC/Patran model. Similar results are reported. High stress region is confined to the tube wall as anticipated. Data presented by both Figures 12 and 13 confirms that the NDE and the finite element findings are in good agreement. However, the advantage of analyzing an NDE produced model versus CAD model is that the NDE offers more details concerning deformities and structural abnormalities which can assist greatly in the structural evaluation of tested components. While in the CAD modeling, all the flaws have to be artificially modeled.

The above information concerning the stress distributions and their magnitude are of high value since they offer a base line design limit that can assist in performing a life estimation process. Furthermore, the work performed represents the beginning of a series of analytical developments and studies to automate the procedure employed in

connecting NDE with finite element. Efforts are ongoing and more representative case studies are expected to be reported in the near future.

Conclusions

Prototyping a 3D volume of a composite structure by means of reading in series of 2D images generated via computed tomography and by using and integrating commercial software, e.g. Velocity^2, Msc/Patran, Hypermesh, was successfully demonstrated. The building process from structural modeling to the analysis level was outlined. Stress analysis of the composite cooling panel under combined thermo-mechanical loading conditions was performed. Major accomplishments are listed below:

1. Rendering of a 3D object from a series of CT scan images using Velocity^2 software is feasible and its complexity increases with the large amount of information provided by the CT data.
2. The analytical results of the high stress regions correlated well with the damage sites as identified by the CT scans.
3. The development of an essential translator code to transfer an STL file into a MSC/Patran Neutral file was accomplished.
4. The ability to read an STL file via HyperMesh utility software was revealed to be useful and to offer a shortcut to generating an IGES file that is suitable for FEA meshing.
5. The procedure applied in constructing a 3D volume remains somewhat cumbersome. A new release of HyperMesh is expected to address most of the NDE-FEA issues by having many automated features that should reduce the number of sequences required. Nevertheless, the current activities are promising and structural assessment of a particular component is achievable.
6. Until more suitable means to extensively reduce the size of the models are established, analyzing an entire component without any alterations in structural details would remain a barrier.

References

1. Griffin, A., McMillin, S. and Knox, C., "Practical Examples: Using Scan Data for Reverse Engineering" AUTOIFACT '97, Nov. 3–6, 1997, Detroit, Michigan.
2. Yancey, R. et al. 1996. "Integration of Reverse Engineering, Solidification Modeling, and Rapid Prototyping Technologies for the Production of Net –Shape Investment Cast Tooling", Advanced Research and Applications Corporation (ARACOR) Dayton, Ohio.
3. Velocity^2, Image Processing and 3D reconstruction visualization software, Version 2.0, July 98, IMAGE3, LLC, Salt Lake City, Utah.
4. Baaklini, G., "Engine Materials Characterization and Damage Monitoring by Using X-Ray Technologies", NASA TP–3328, 1993.
5. MARC general purposes Finite Element Analysis program, 1996, Vol. A: User Information manual; Vol. F: Theoretical Manual. MARC Analysis Research Corporation, Palo Alto, California.
6. MSC/PATRAN Graphics and Finite Element Package. Vols. I and II. The MacNeal-Schwendler Corporation, 1997, Costa Mesa, California.
7. VoxBlast, Ver. 1.4, 1998, VayTek, Inc., Iowa City, Iowa.
8. Corel Photopaint, Ver. 5.00 G1, 1988 Corel Corporation, Ottawa, Ontario, Canada.
9. Adobe Photoshop, Version 5.02 Adobe System, Inc., San Jose, California.
10. Altair HyperMesh, Ver. 3.1, 1999, Altair Engineering, Inc., Troy, Mi.
11. Geomagic Decimator 1.0.2, Rain Drop Geomagic, Inc., Research triangle park, NC.
12. Jeffery Stout, Personal Communication, Boeing Aerospace Company, Seattle.
13. Advanced Research & Applications Corp., ARACOR, "Nondestructive Tomograohy Acquisition of Stiffness and Density Information for Finite Element Analysis", Final Report, NAS3–97016, 1999, Dayton, Ohio.
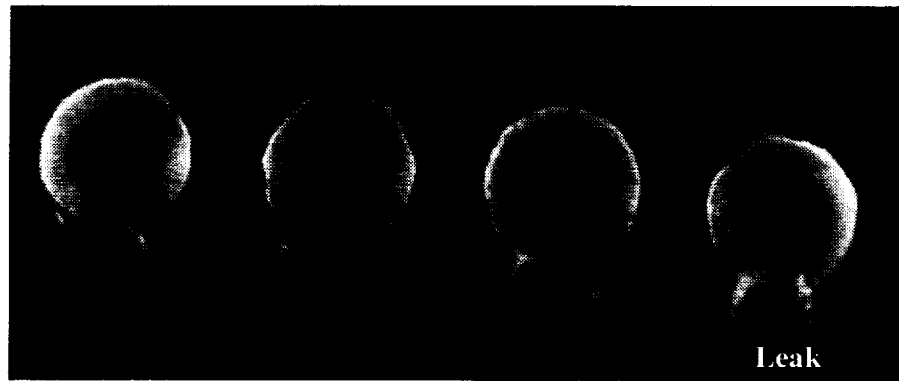
5 mm

Figure 1. Selected CT slice of the cooling panel.
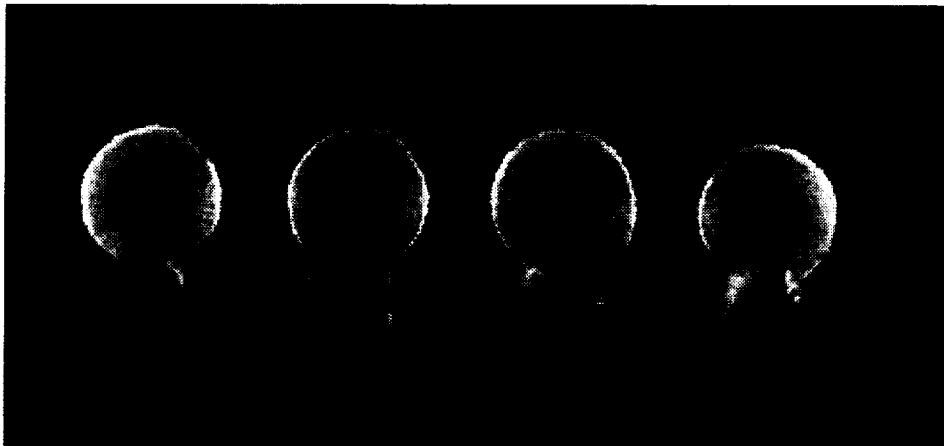
5 mm

Figure 2. Pre-processed and filtered image of the CT slice in Fig.1.
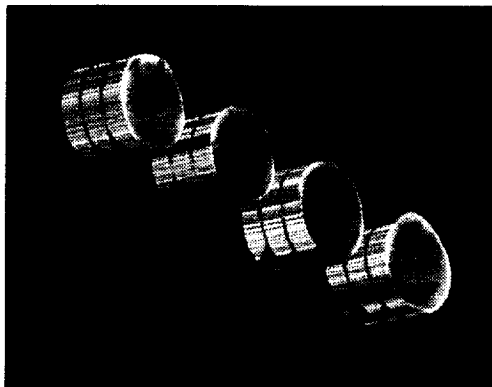

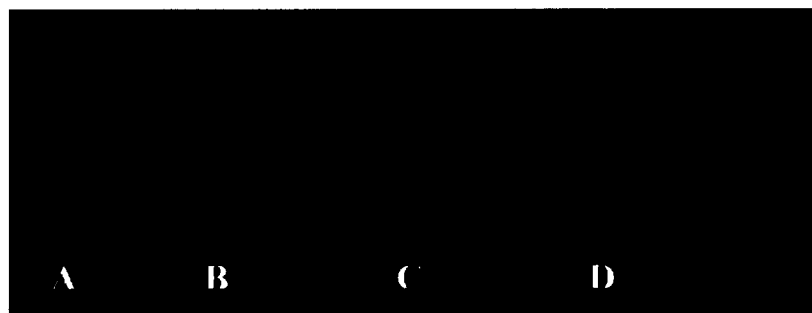**Figure 3. 3D Volume rendering of a set of fifty 2D filtered CT slices.**

Figure 4. Velocity^2 polygon rendering format of the same set of slices in Fig.3.
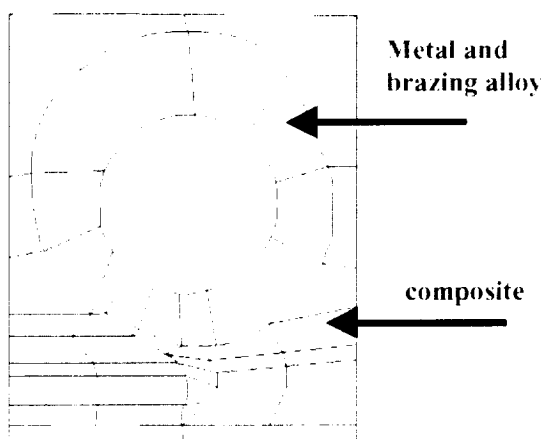

Figure 5. Polygon wiremesh of tube A in Fig.4.

Metal and brazing alloy

composite

Figure 6. Hypermesh IGES model extracted from Fig.5 modeling tube A and composite.
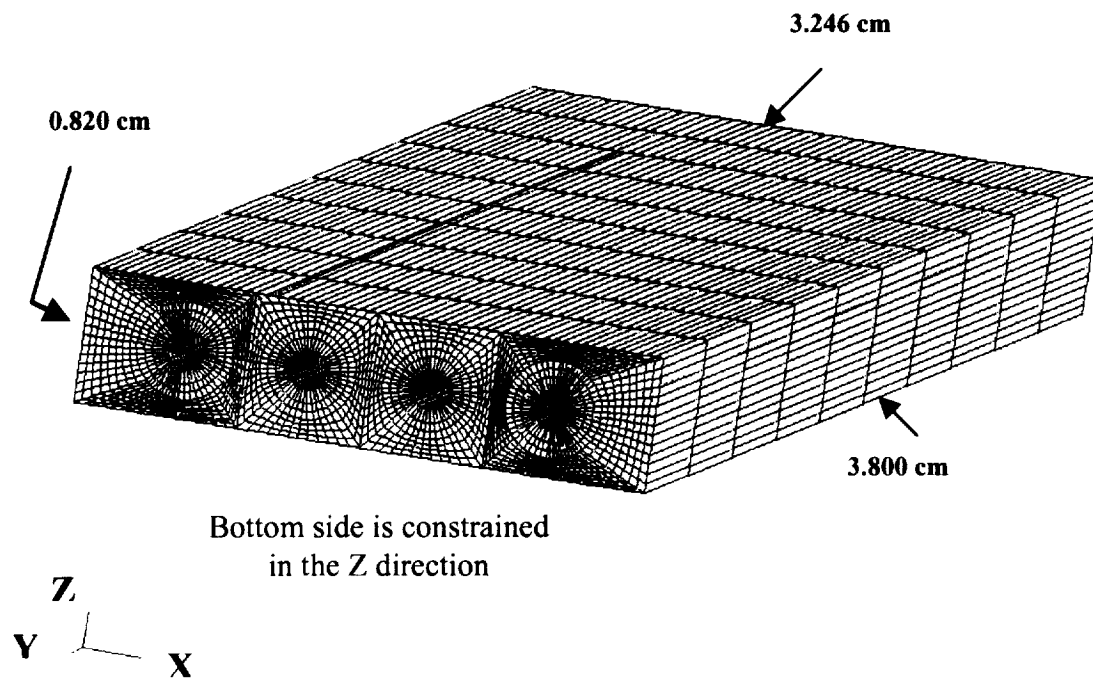
Figure 7. MSC/Patran finite element model of the panel with cooling tubes.
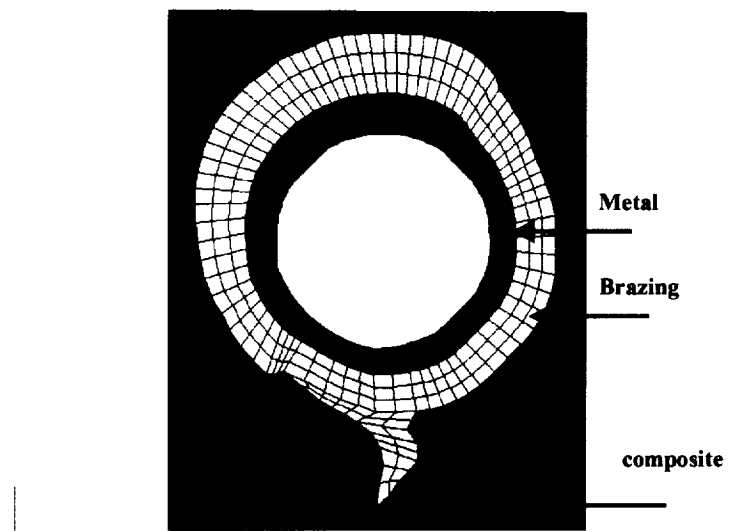
Figure 8. Materials distribution in the panel.
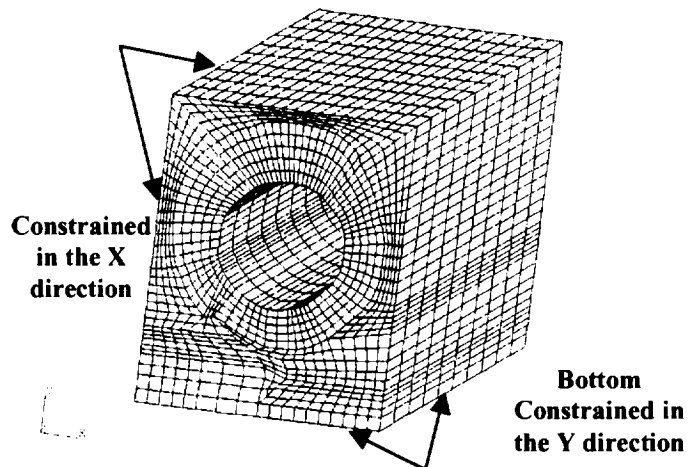
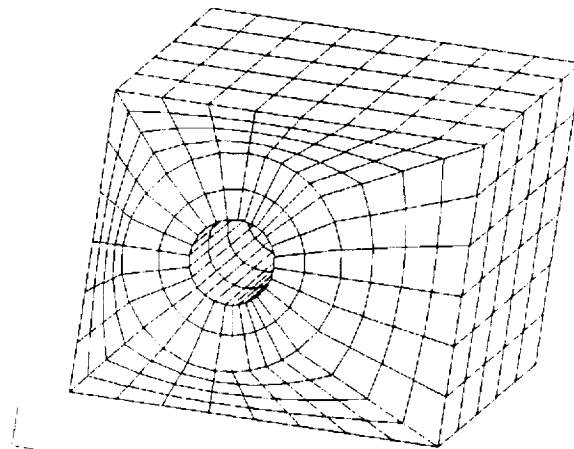Figure 9. Finite element model of tube A built from the IGES file of the CT data.

Figure 10. MSC/Patran generated finite element model of tube A
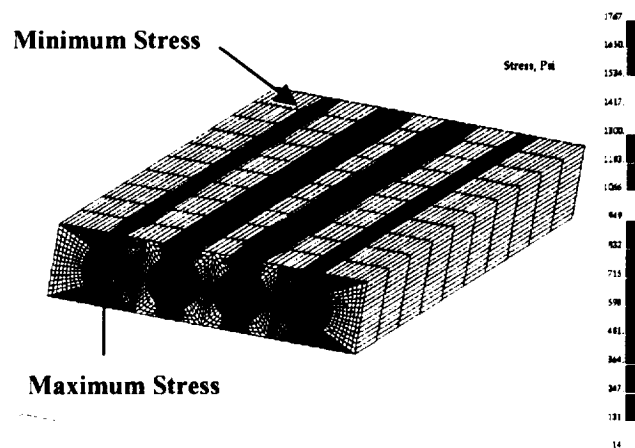and composite with similar constrains to Fig.9.

Figure 11. Von Mises stress distribution for the cooling panel based on CT data.
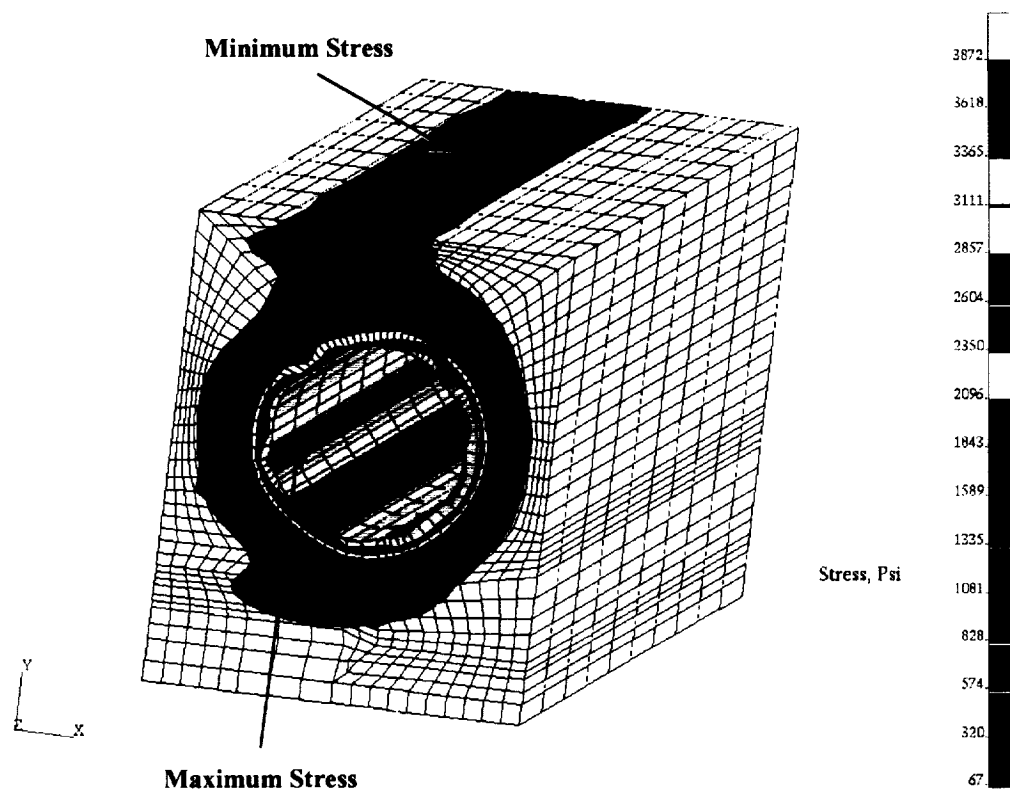
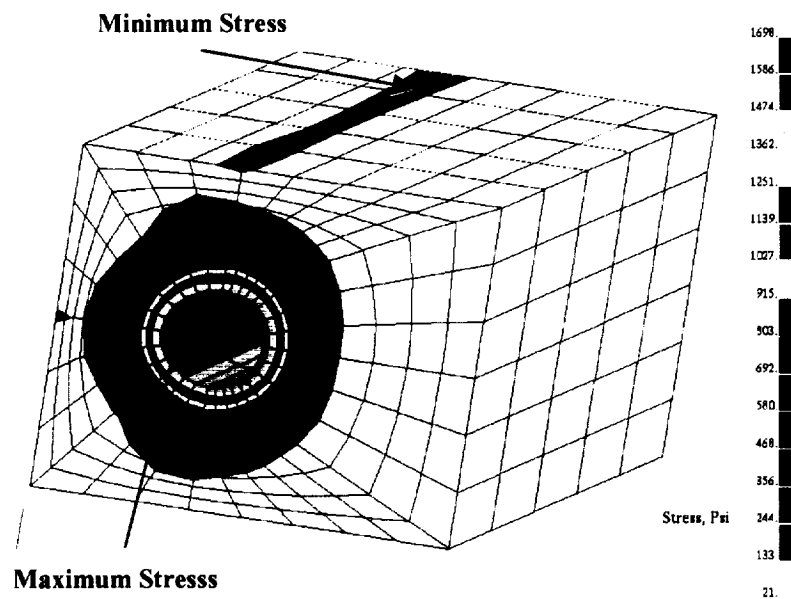**Figure 12. Von Mises stress distribution for tube A and surrounding based on CT data.**

Figure 13. Von Mises stress distribution for tube A and surrounding based on MSC/Patran data.

REPORT DOCUMENTATION PAGE

Public reporting burden for this collection of information is estimated to average 1 hour per response, including the time for reviewing instructions, searching existing data sources, gathering and maintaining the data needed, and completing and reviewing the collection of information. Send comments regarding this burden estimate or any other aspect of this collection of information, including suggestions for reducing this burden, to Washington Headquarters Services, Directorate for Information Operations and Reports, 1215 Jefferson Davis Highway, Suite 1204, Arlington, VA 22202-4302, and to the Office of Management and Budget, Paperwork Reduction Project (0704-0188), Washington, DC 20503.

1. AGENCY USE ONLY (Leave blank)	2. REPORT DATE April 2000	3. REPORT TYPE AND DATES COVERED Technical Memorandum

4. TITLE AND SUBTITLE

Challenges in Integrating Nondestructive Evaluation and Finite Element Methods for Realistic Structural Analysis

5. FUNDING NUMBERS

WU–523–31–13–00

6. AUTHOR(S)

Ali Abdul-Aziz, George Y. Baaklini, Dmitri Zagidulin, and Richard W. Rauser

7. PERFORMING ORGANIZATION NAME(S) AND ADDRESS(ES)

National Aeronautics and Space Administration
John H. Glenn Research Center at Lewis Field
Cleveland, Ohio 44135–3191

8. PERFORMING ORGANIZATION REPORT NUMBER

E–12208

9. SPONSORING/MONITORING AGENCY NAME(S) AND ADDRESS(ES)

National Aeronautics and Space Administration
Washington, DC 20546–0001

10. SPONSORING/MONITORING AGENCY REPORT NUMBER

NASA TM—2000-209952

11. SUPPLEMENTARY NOTES

12a. DISTRIBUTION/AVAILABILITY STATEMENT

Unclassified - Unlimited
Subject Category: 20   Distribution: Nonstandard

This publication is available from the NASA Center for AeroSpace Information, (301) 621–0390.

12b. DISTRIBUTION CODE

13. ABSTRACT (Maximum 200 words)

Capabilities and expertise related to the development of links between nondestructive evaluation (NDE) and finite element analysis (FEA) at Glenn Research Center (GRC) are demonstrated. Current tools to analyze data produced by computed tomography (CT) scans are exercised to help assess the damage state in high temperature structural composite materials. A utility translator was written to convert velocity (an image processing software) STL data file to a suitable CAD-FEA type file. Finite element analyses are carried out with MARC, a commercial nonlinear finite element code, and the analytical results are discussed. Modeling was established by building MSC/Patran (a pre and post processing finite element package) generated model and comparing it to a model generated by Velocity in conjunction with MSC/Patran Graphics. Modeling issues and results are discussed in this paper. The entire process that outlines the tie between the data extracted via NDE and the finite element modeling and analysis is fully described.

14. SUBJECT TERMS

Finite element analysis; Computed tomography; Image processing; Stress analysis; Nondestructive evaluation

15. NUMBER OF PAGES

18

16. PRICE CODE

A03

17. SECURITY CLASSIFICATION OF REPORT Unclassified	18. SECURITY CLASSIFICATION OF THIS PAGE Unclassified	19. SECURITY CLASSIFICATION OF ABSTRACT Unclassified	20. LIMITATION OF ABSTRACT

NSN 7540-01-280-5500